ТЕОРИЯ КОРАБЛЯ И СТРОИТЕЛЬНАЯ МЕХАНИКА

Научная статья
УДК 532.582
https://doi.org/10.24866/2227-6858/2025-2/15-25

Анализ влияния моделей турбулентности, топологии сетки и характеристик потока на точность прогнозирования гидродинамических характеристик изолированного гребного винта

Рами Мамдух Али✉**, Мохаммед Аднан Ризк**

Санкт-Петербургский государственный морской технический университет,
Санкт-Петербург, Российская Федерация
✉ ramimamdouhali@gmail.com

Аннотация. В работе рассматривается влияние модели турбулентности, топологии сетки и характеристик потока на точность численного прогнозирования гидродинамических характеристик гребного винта в широком диапазоне относительной поступи. Уравнения RANS решены с помощью Ansys Fluent для оценки гидродинамических характеристик модели изолированного гребного винта типа E779A. Достоверность численных результатов была проверена известными экспериментальными данными; численно-экспериментальное сравнение показало хорошее соответствие. Были представлены и проанализированы значения коэффициентов упора, момента и КПД для различных моделей турбулентности и топологий сетки как в стационарном, так и в нестационарном потоке. Установлено, что использование переходной модели турбулентности в сочетании с гексаэдрической сеткой гарантирует высокий уровень точности определения гидродинамических характеристик изолированного гребного винта.

Ключевые слова: гребной винт, INSEAN E779A, CFD, RANS, Ansys Fluent, модели турбулентности, испытания изолированного гребного винта

Для цитирования: Али Р.М., Ризк М.А. Анализ влияния моделей турбулентности, топологии сетки и характеристик потока на точность прогнозирования гидродинамических характеристик изолированного гребного винта // Вестник Инженерной школы Дальневосточного федерального университета. 2025. № 2(63). С. 15–25.

SHIP THEORY AND STRUCTURAL MECHANICS

Original article

Analysis of the impact of turbulence models, grid topology and flow characteristics on the precision of propeller's hydrodynamic performances predictions in open water tests

Rami M. Ali ✉**, Mohammed A. Rizk**

Saint Petersburg State Marine Technical University, Saint Petersburg, Russian Federation
✉ ramimamdouhali@gmail.com

Abstract. The effects of the turbulence model, grid topology, and flow characteristics on the accuracy of numerical prediction of propeller hydrodynamic performance were studied over a wide range of advance ratios. RANS equations were solved with Ansys Fluent to evaluate the hydrodynamic coefficients of the E779A propeller model in the open water tests. The validity of the numerical results was examined by a published experimental benchmark; the numerical-experimental comparison revealed a good agreement. Thrust coefficient, torque, and efficiency were presented and analysed for different turbulence models and grid topologies in both steady and unsteady flow. It is found that using the transition turbulence model in conjunction with the hexahedral grid guarantees a high level of accuracy of the propeller hydrodynamic performance in the open water tests.

Keywords**:** marine propeller, INSEAN E779A, CFD, RANS, Ansys Fluent, turbulence models, open water tests

For citation: Ali R.M., Rizk M.A. Analysis of the impact of turbulence models, grid topology, and flow characteristics on the precision of propeller's hydrodynamic performances predictions in open water tests. *FEFU: School of Engineering Bulletin*, 2025, no. 2(63), pp. 15–25. (In Russ.).

Introduction

The hydrodynamic performance of the propeller has been assessed by researchers using a variety of methods. The initial approaches were experimental methods and empirical models. Later, many numerical methods based on potential theory were developed, including the Blade Element Theory (BET), Blade Element Momentum Theory (BEMT), Integral Boundary Layer Method (IBLM), and Boundary Element Method (BEM). As the Computational Fluid Dynamics (CFD) has advanced and computer performance has continued to rise, viscous solutions have grown in popularity and versatility. However, many factors, including CAD geometry, topology, size of computational domain, meshing strategy, and physical modelling, greatly influence the results obtained by the viscous method. As a result, many researchers are interested in precisely determining how these factors affect the propeller's hydrodynamic properties.

The choice of the grid topology affects both the solver's capabilities and the accuracy of the results, so it's crucial to select the appropriate topology for the problem being addressed. Sikirica A. et al assessed the suitability of the hybrid and hexahedral grids for forecasting maritime propeller performances [1]. They evaluated the hydrodynamic properties of PPTC using SST k-ω and realizable k-ϵ turbulent models. The numerical computations were performed with STAR-CCM+ and Ansys Fluent. According to their findings, both hexahedral and hybrid grids yield comparable results; however, when compared to SST k-ω, the realizable k-ϵ models yield more accurate results, particularly at high advance ratios J. Morgut and Nobile also investigated the impact of mesh type and turbulence model on the accuracy of propeller performances [2]. They studied the hydrodynamic performance of two different propellers, E779A and P5168 types. Two different mesh types were used: hybrid mesh and hexahedral mesh. The flow around the propellers was computationally modeled with the CFX software using two turbulence models, SST and RSM. They reported that the different meshes and turbulence models used in their research had shown comparable levels of accuracy. Tu T.N, in his work, used two different grids: hexahedral and polyhedral, and two turbulence models to examine the PPTC propeller's hydrodynamic performances in open water tests [3]. The numerical results of the RANS (Reynolds-averaged Navier-Stokes equations) solver indicated that both turbulence models have a similar degree of accuracy, but the hexahedral mesh produced slightly more accurate results than the polyhedral. Wang and Walters compared the hydrodynamic properties of the propeller using the SST k-ω turbulence model and the transition-sensitive turbulence model TSM [4]. Their results indicated that the calculated thrust with the TSM model showed improvement compared to the SST model, but the relative errors remain large when compared to experimental values. However, other authors' works, including those of [2], [5] and [6], also examined the impact of the turbulence model and reported that the effectivity of turbulence models depends directly on the advance ratio.

Many researchers have used the viscous model to evaluate the propeller’s hydrodynamic performance [7–11] but most studies used two-equation turbulence models and were limited to comparing only two models - one of which was commonly SST k-omega turbulence model. Despite its advantages, in terms of adaptability to complicated surfaces like propellers, the tetrahedral grid has received insufficient attention and comparison. The steady-state assumption, commonly used in propeller performances prediction, may not capture all dynamic interactions, so it is worth comparing it to the unsteady-state assumption.

In this paper, the viscous flow around a four-blade propeller was simulated over a wide range of advance ratios. Various turbulence models were implemented, effects of steady and unsteady assumptions on the propeller performance were investigated and analysed across various grid topologies.

Problem definition

Propeller geometry

This study will employ the E7794 propeller to numerically simulate the flow and explore the propeller's hydrodynamic properties in open water tests. E7794 is a four-blade, fixed-pitch, low-skew propeller designed in 1959 by the Italian Ship Model Basin (INSEAN) for hydrodynamic and hydro-acoustic experiments. It is well documented in the literature, making it a benchmark for confirming numerical results. Experimental data used throughout the course of this study were carried out in INSEAN towing tank and publicly accessible under the name INSEAN E779A Propeller Experimental Dataset [12]. However, the propeller shape and physical attributes are depicted in the (Fig.) and (Table 1), respectively.

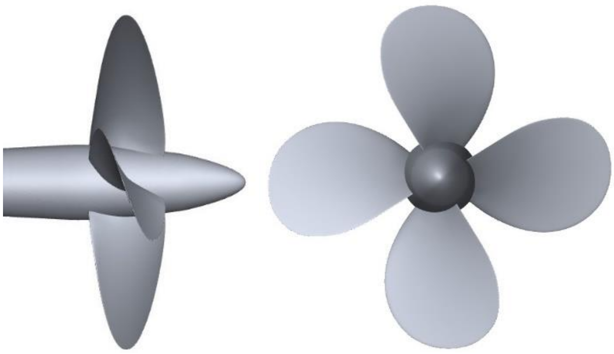


Fig. 1. The geometry of E7794 propeller

Table 1

Principal particulars of E7794 propeller

Diameter	D	m	0.2273
Expanded Area	A_e/A_0	-	0.6890
Nominal Pitch	P	m	0.2500
Pitch Ratio	P/D	-	1.1000
Number of Blades	Z	-	4
Boss Ratio	r_h/R	-	0.2000
Skew Angle at Blade Tip	θ_s^{tip}	deg	4.3500
Hub Diameter	D_h	m	0.0455
Hub Length	L_h	m	0.0683
Reference chord	c	m	0.0870
Nominal Rake	i	deg	4.5833

Nondimensiolity

The thrust T, torque Q, and efficiency of the propeller are expressed in terms of dimensionless coefficients K_T, K_Q, and η_0, as follows:

$$K_T = \frac{T}{\rho n^2 D^4}, \tag{1}$$

$$K_Q = \frac{Q}{\rho n^2 D^5}, \tag{2}$$

$$\eta_0 = \frac{K_T}{K_Q} \cdot \frac{J}{2\pi}, \tag{3}$$

where D is the diameter of the propeller, n is the rotational speed of the propeller and ρ is the density of the fluid. J is the advance ratio that describe the relationship between the speed at which the ship is moving forward V_A and the speed at which its propeller is turning, J is given as follows:

$$J = \frac{V_A}{nD}, \tag{4}$$

Governing equations

Navier-Stokes and continuity equations provide a complete mathematical description of the flow of incompressible Newtonian fluids, the differential forms of these equations are as follows:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} = \rho g_i - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial {x_j}^2}, \tag{5}$$

$$\frac{\partial(u_i)}{\partial x_i} = 0, \tag{6}$$

Where μ is the dynamic viscosity, p is the pressure, u_i is the velocity vector and $x_{i,j}$ is the spatial vector.

These equations are discretized over the domain using the well-known finite volume method FVM. To characterize the turbulent flow around the propeller in a way that achieves a balance between accuracy and computational cost, RANS is used. According to RANS, averaged Navier-Stokes's equations for incompressible flow are written as follows:

$$\frac{\partial(\rho \overline{u_i})}{\partial t} + \frac{\partial(\rho \overline{u_j u_i})}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} - \rho \overline{u_i' u_j'}\right), \tag{7}$$

Where: τ_{ij} is the viscous stress, while $\rho \overline{u_i' u_j'}$ is the Reynolds stress.

The appearance of Reynolds stress makes the previous set of equations unclosed and unsolvable set. To close this system of equations, it is necessary to model the turbulence and estimate the value of τ. In this study, various turbulence models will be used to close the system, and their impact on the prediction of propeller's hydrodynamic performance will be evaluated and compared. Turbulence models intended to be studied are listed and classified here according to their number of equations: one equation (Spalart-Allmaras), two-equation (RNG k-epsilon, realized k-epsilon, SST k-omega), and a three-equation model (transition k-kl-omega).

Numerical formation

Computational domain

A cylindrical domain with a diameter of 5D was used to simulate the flow around the propeller; the inlet and outlet were placed at a distance of 3D upstream and 5D downstream from the propeller plan, respectively. To handle the propeller rotation, the domain is equipped with a coincided cylinder that encloses the propeller disc and forms the rotary domain.

Grid generation

Three types of mesh; Hexahedral, Tetrahedral and poly-hexcore mesh were generated by Ansys Fluent Meshing tool. The non-dimensional wall distance y^+ for all typologies of grids was kept unchanged, the average value of y^+ on solid surfaces is 6. $y^+ = u_t y/v$, where u_t is the friction velocity, y is the distance to the wall, and v is the kinematic viscosity of the fluid. However, to accurately capture the flow characters fine zones surrounds propeller tips and hub were added.

The computational domain is presented in (Fig. 1), a general view highlighting different zones of the computational domain is shown in (Fig. 2).

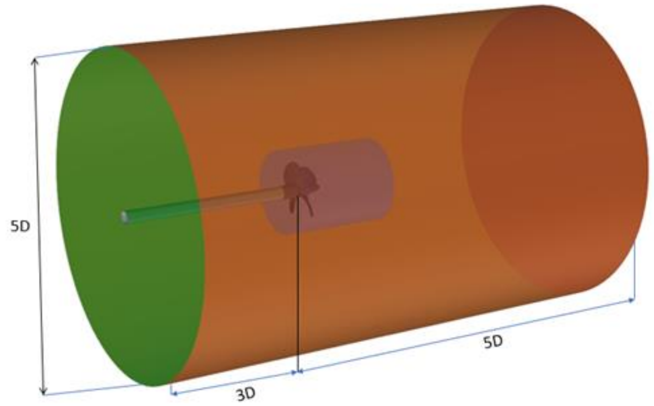


Fig. 1. General view of the computational domain

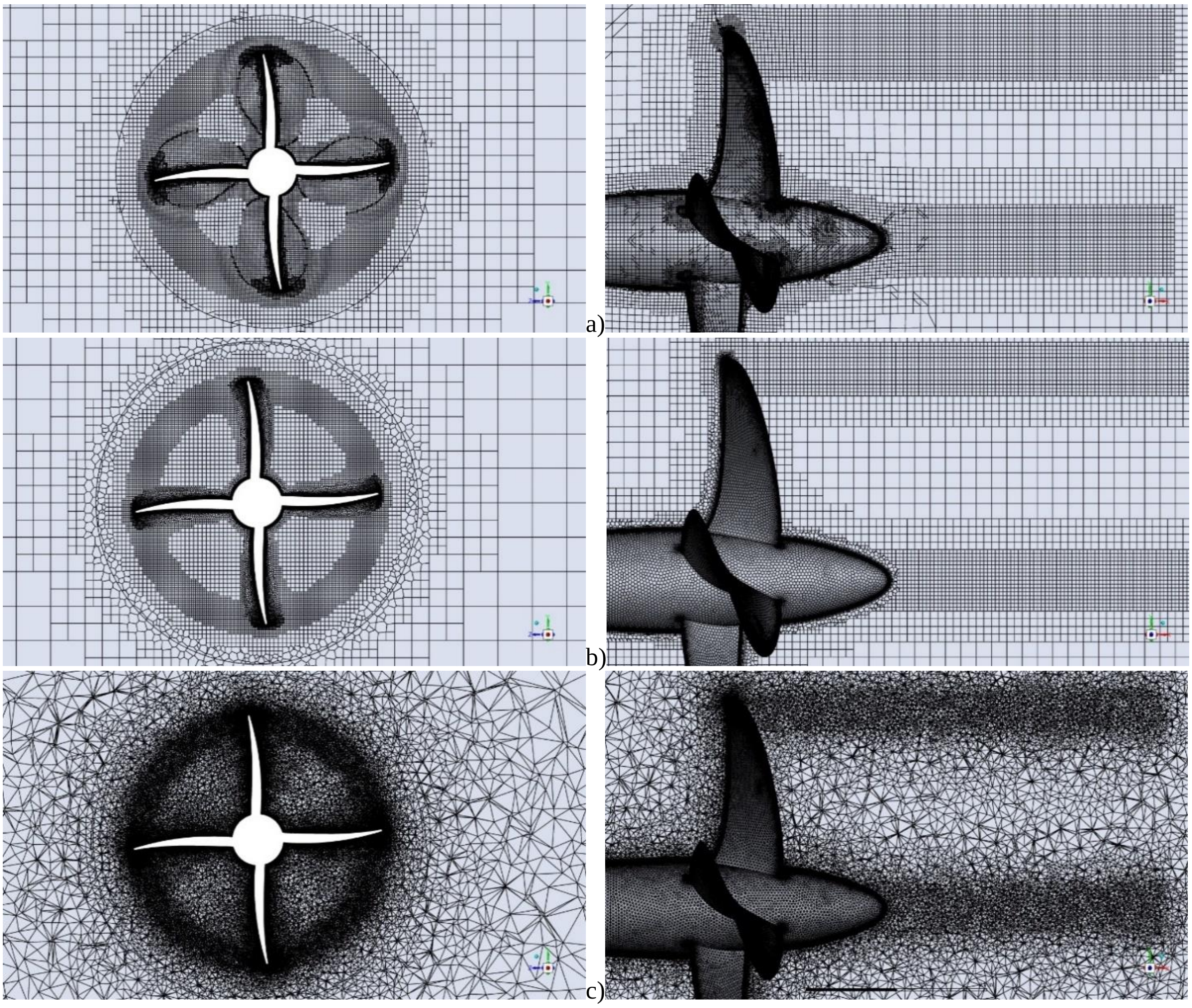


Fig. 2. Overview of the computational domain and Grid structure around E779A propeller: (a) hexahedral mesh; (b) poly-hexcore mesh; (c) tetrahedral mesh.

Numerical settings

Finite Volume Method (FVM) is used to discrete the flow domain into a finite number of control volumes. Ansys fluent is used to simulate the viscous flow for propeller open water tests and solve the Reynolds-averaged Navier-Stokes (RANS) equations.

Simulations were conducted over a range of advance coefficients $J \ni [0\sim 1.145]$. corre-sponded to the range of Reynolds numbers $[462647 - 520462]$. The Reynolds number was defined as $Re = \frac{C_{0.7}\sqrt{V_A{}^2+(0.7\pi nD)^2}}{v}$, where $C_{0.7}$ was the propeller blade chord at 70% of radius, D and n is the diameter and rotational speed propeller, respectively. v is the kinematic viscosity.

The propeller rotation is simulated using the moving reference frame (MRF) approach, rotational speeds were set at $n = 11.7881\ [rps]$. Turbulence intensity Tu and eddy viscosity ratio μt/μ are set to 2% and 10 respectively [13]. The time step for unsteady simulations was set to $\Delta t = 2.35 \cdot 10^{-4}\ [s]$ corresponds to one degree of propeller rotation.

According to the environment of the experimental work, the water condition is modeled as fresh water at 16℃, the corresponding value of density ρ and kinematic viscosity ν are set to $\rho = 102.06\ [KgS^2/m^4]$ and 1.1099 $10^6\ [m^2/S]$, respectively.

Uniform flow velocity condition is set on the inlet, a Slip conditions to the stator surface and no-Slip condition was imposed to Shaft, hub and blade walls. On the outlet zero Pa static pressure was imposed.

Grid convergence study

Seven successive poly-hexcore grids with a refinement factor $r_G = \sqrt{2}$ and average $y+ \approx 6$ were created. Grid convergence was studied at $J = 0.747$ using steady simple solver and SST k-omega turbulence model. Numerical results and experimental data were compared, grid sizes and relative errors are provided in (Table 2). According to the presented results, the grid “6” will be used.

Table 2

Convergence study results with SST k-omega turbulence model at j =0.747

i	Cell counts	$r_G=\sqrt{2}$	Kt CFD	Kq CFD	η CFD	Kt Exp	Kq Exp	η Exp	Error kt%	Error kq%	Error Eta%
1	951542	8.00	0.2083	0.03974	0.623	0.222	0.041	0.652	6.152	1.886	4.3472
2	1064283	5.66	0.2088	0.03974	0.625				5.905	1.889	4.0938
3	1269072	4.00	0.2088	0.03964	0.626				5.944	2.121	3.9054
4	1444123	2.83	0.2094	0.03971	0.627				5.662	1.960	3.7761
5	2096776	2.00	0.2098	0.03981	0.627				5.473	1.714	3.8249
6	2572131	1.41	0.2099	0.03985	0.626				5.442	1.599	3.9052
7	6087811	1.00	0.2100	0.03986	0.6265				5.423	1.580	3.9049

Results

Effect of turbulent model

The propeller's hydrodynamic performances are investigated using a steady-state solver over a poly-hexcore mesh (hybrid). In (Fig. 3) the experimental results are compared with the numerical results obtained by two-equation turbulence models: RNG, Realizable k-Epsilon, and KW SST turbulence models.

In general, the numerical results obtained with RNG and Realizable k-Epsilon turbulence models agree well with the experimental values of K_T and K_Q over the entire range of the examined advance ratio j except for the narrow range $j \ni [0\sim 0.149]$. Outside of this range, both models slightly overestimate the torque coefficient K_Q and underestimate the thrust coefficient K_T. The numerical results obtained by KW SST are directly related to the advance coefficient j. At low values of j, KW SST accurately predicts the thrust coefficient K_T while slightly overestimates the torque coefficient K_Q. At high values of j, it slightly underestimates both K_Q and K_T. However, comparing the numerical results of all analysed two-equation models reveals that KW SST better predicts K_T and K_Q values across the whole range of j.

Regarding efficiency, the investigated tow-equation turbulence models accurately predict the efficiency at low advance ratios, but clearly underestimate it at high advance ratios. However, the

KW SST turbulence model predicts efficiency more accurately than the other models studied over the entire values of j, except for $j = 1.145$ where realizable k-Epsilon model is better. The error in evaluating propeller efficiency at $j = 0.845$ reaches -9.3% for RNG k-ε, -8.1% for realizable k-ε, and only -6.1% for SST k-ω model.

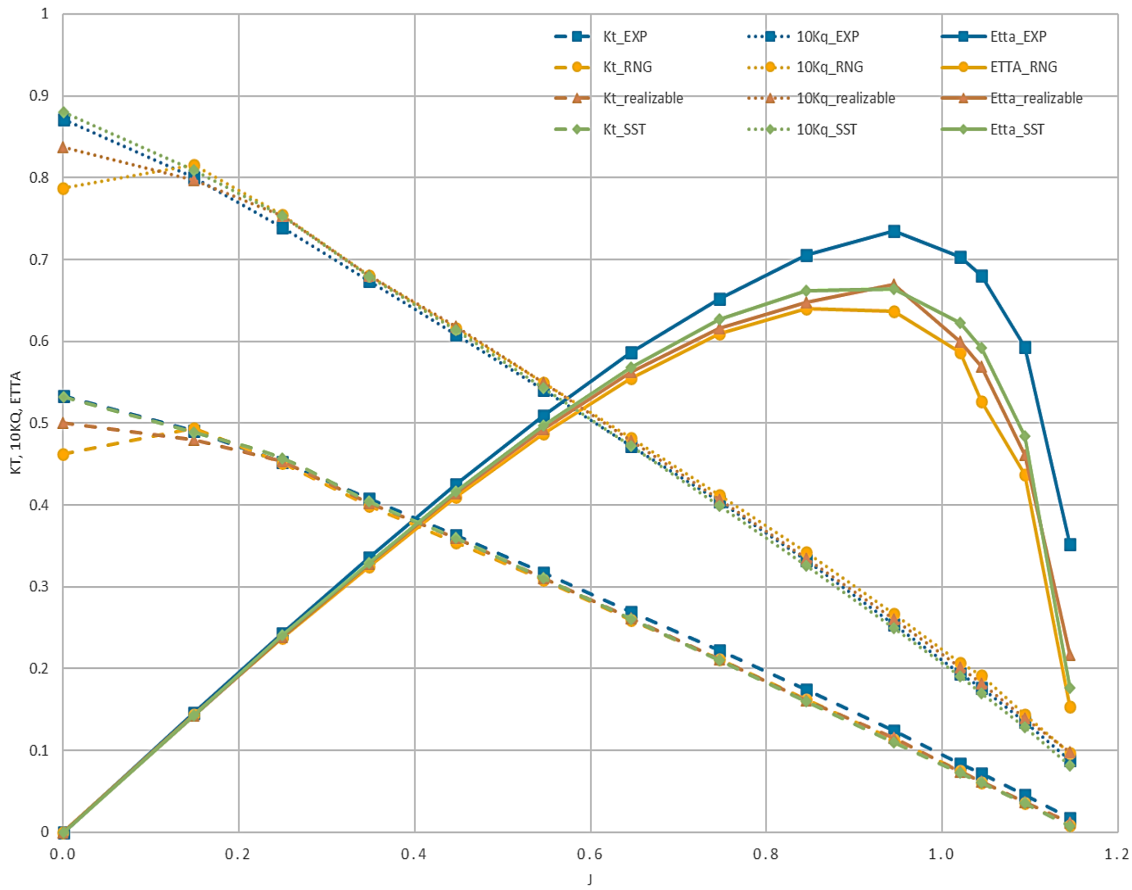


Fig. 3. Propeller hydrodynamic curves using two-equation turbulence models. Hybrid mesh, steady state.

To investigate the effect of the turbulence model's number of equations on the numerical prediction accuracy, one-, two- and three-equation models were used. The results were validated against experimental data in (Fig. 4).

The numerical results of the studied turbulence models generally agree well with the experimental results for the thrust and torque coefficients over the entire range of the advance ratio j, with the exception of the narrow range of $j \ni [0\sim 0.149]$, where one- and three-equation models (Spalart-Allmaras and k-kl-omega, respectively) significantly underestimate both the thrust and torque coefficients. However, outside this range, the one-equation model (Spalart-Allmaras) underestimates the thrust coefficient while overestimates the torque coefficient. Both two- and three-equation models (KW SST and k-kl-omega, respectively) accurately predict the thrust coefficient, but their ability to predict the torque coefficient varies with the advance ratio, since both overestimate the torque coefficient at low advance ratios and underestimate it at high advance ratios.

Regarding efficiency; the studied models accurately predict propeller efficiency η at low advance ratios j, while at high advance ratios the discrepancy between numerical and experimental values increases significantly. (Fig. 4) shows that the relative error committed in evaluating the efficiency at advance ratio $j = 0.945$ is approximately -13.2% for the one-equation model, -9.6% for the two-equation model, and 3.1% for the three-equation model. However, curves of propeller efficiency also indicate an improvement in the estimation ability as the number of equations of the turbulence model increases.

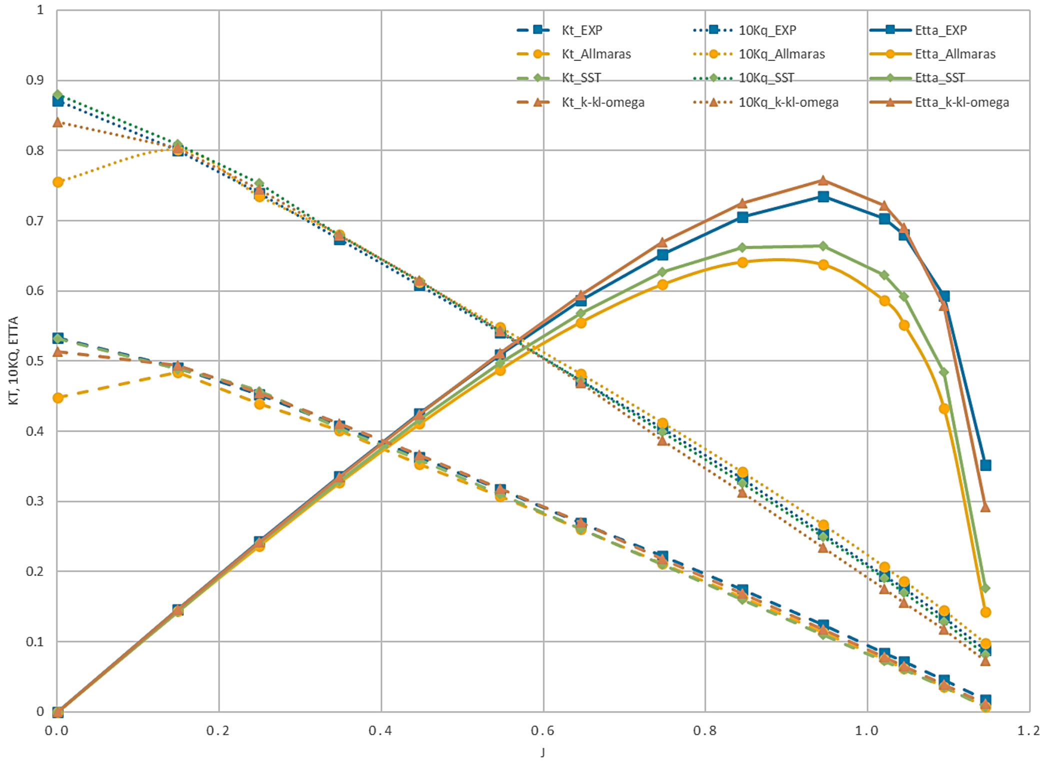


Fig. 4. Propeller hydrodynamic curves using different type of turbulence models. Hybrid mesh, steady state.

Effect of mesh topology

To investigate the effect of mesh topology on propeller hydrodynamic performance prediction, a steady-state numerical simulation is performed using a three-equation turbulence model (k-kl-omega) and three distinct mesh topologies: hexahedral, tetrahedral, and poly-hexcore (Fig. 6).

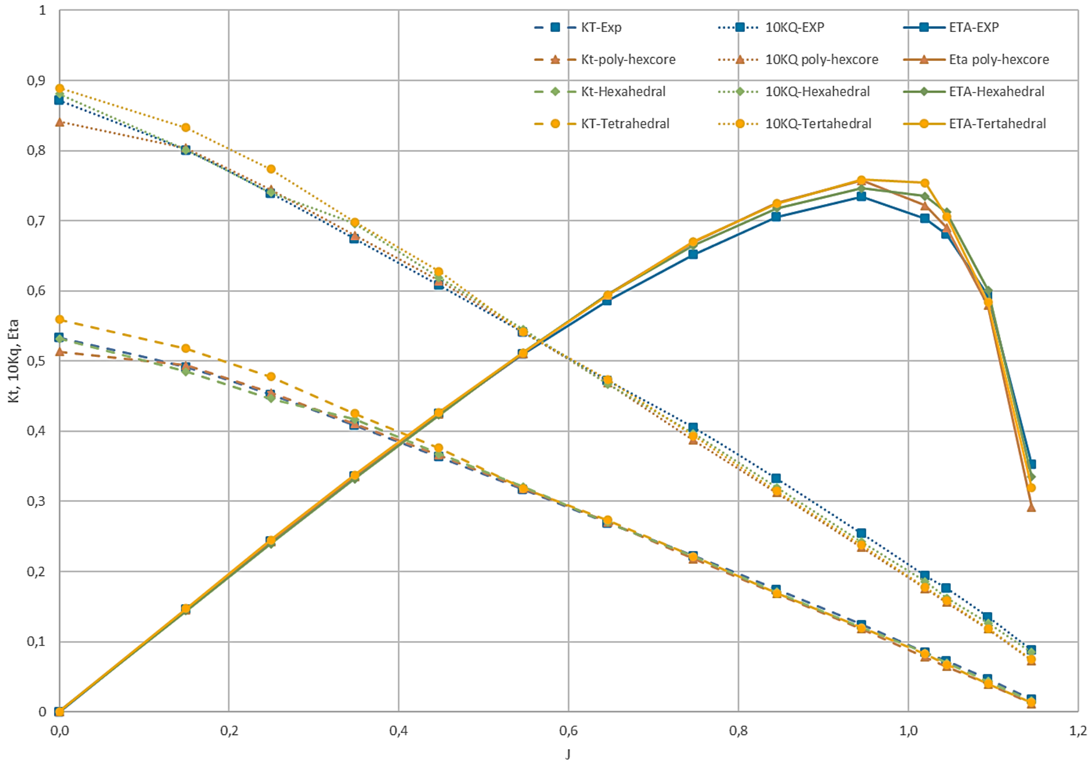


Fig. 5. Propeller hydrodynamic performance using different grid topology, transition turbulence model, steady state.

Tetrahedral mesh clearly overestimates thrust and torque coefficients at low advance ratios while underestimates them at high advance ratios, the tetrahedral mesh yields the least accurate findings among the topologies tested.

The hexahedral and poly-hexcore topologies produce similar outcomes and are more consistent with experimental results than the tetrahedral mesh. However, numerical results from poly-hexcore meshes correspond better with experimental data at low advance ratios, but hexahedral meshes are more accurate at higher advance ratios.

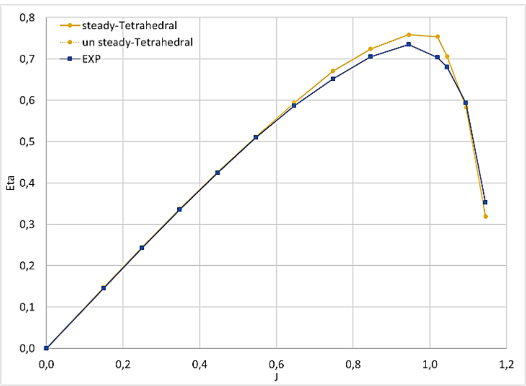
Regarding efficiency, hexahedral mesh outperforms tetrahedral and poly-hexcore meshes across the whole examined range of advance ratios, with the exception of $j \ni [1.02\sim 1.045]$, where poly-hexcore mesh appears to be more accurate.

At $j = 0,945$, the error in evaluating propeller efficiency reaches 3.24% for tetrahedral, 3.08% for poly-hexcore and only 1.60% for the hexahedral mesh.

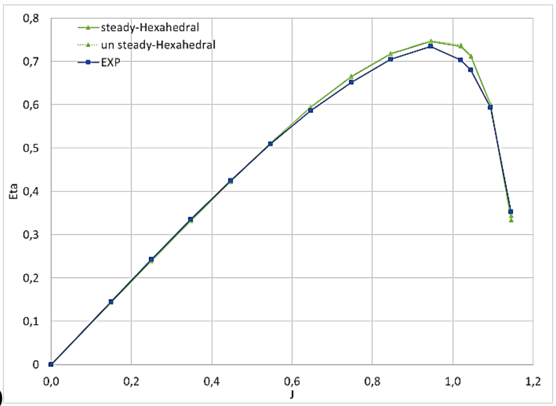
Effect of steady and unsteady assumptions on the propeller's hydrodynamic performances

SIMPLE unsteady solver was used to study the effect of steady and unsteady assumption on the numerical prediction of propeller's hydrodynamic performance. Investigation was carried out using three-equation turbulence model (k-kl-omega) on hexahedral mesh. However, to generalize results, the simulations were repeated on different mesh topologies (tetrahedral and poly-hexcore) and presented in (Fig. 6).

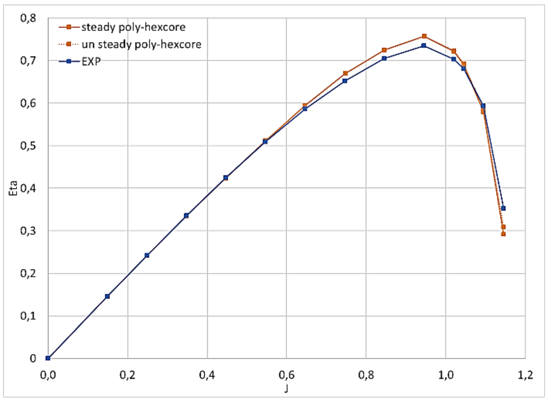
(Fig. 6) indicates that using unsteady assumptions did not improve the numerical simulation of the tetrahedral mesh at all, but did slightly improve the hexahedral and poly-hexcore mesh results at high advance ratios. Using unsteady solver reduced the inaccuracy at $j = 1.094$ from 1.29% to -1.09% for hexahedral mesh and from -2.38 to -1.02% for poly-hexcore mesh.



a)



b)



c)

Fig. 6. steady and unsteady evaluation of propeller performance using transition turbulence model and different grid topology.

Conclusion and recommendations

A series of systematic computations were performed to investigate the influence of the grid topology, turbulence model, and steadiness assumption on the accuracy of propeller hydrodynamic performance prediction over a wide range of advance ratios j. Ansys fluent was used to conduct numerical simulations on the E779A propeller model. The propeller hydrodynamic coefficients K_T, K_Q, and η were evaluated in open water tests using RANS in both steady and unsteady approximations. The turbulent models under consideration included (Spalart-Allmaras, RNG k-epsilon, realized k-epsilon, SST k-omega, and transition k-kl-omega). The simulations used three distinct grid topologies (hexahedral, tetrahedral, and poly-hexcore).

The results obtained in this work can be summarized as follows:

- Among the analysed two-equation models (RNG k-epsilon, realized k-epsilon, and SST k-omega), the SST k-omega turbulence model had the best ability to predict the propeller's hydrodynamic performances over the entire range of studied advance ratios.
- When more equations are used in turbulence models, the numerical results become more accurate.
- The transition three-equation turbulence model (k-kl-omega) predicts propeller performance better than the one- and two-equation models evaluated.
- Hexahedral grids outperform poly-hexcore and tetrahedral grids in terms of properly predicting propeller performances at various advance ratios.

Unsteady assumptions had no impact on the numerical simulation of the tetrahedral mesh, but they did slightly improve the hexahedral and poly-hexcore mesh results only at high advance ratios.

ВКЛАД АВТОРОВ | CONTRIBUTION OF THE AUTHORS

Р.М. Али, М.А. Ризк – анализ и интерпретация результатов; подготовка и редактирование текста. Все авторы прочитали и одобрили окончательный вариант рукописи.

R.M. Ali, M.A. Rizk – analysis and interpretation of results; draft manuscript preparation. All authors reviewed the results and approved the final version of the manuscript.

КОНФЛИКТ ИНТЕРЕСОВ | DISCLOSURE

Авторы заявляют об отсутствии конфликта интересов.
The authors declare no conflict of interest.

СПИСОК ИСТОЧНИКОВ

1. Sikirica A., Čarija Z., Kranjčević L., Lučin I. Grid type and turbulence model influence on propeller characteristics prediction // Journal of Marine Science and Engineering. 2019. Vol. 7. № 10. Art. 374. DOI: https://doi.org/10.3390/jmse7100374.
2. Morgut M., Nobile E. Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow // Ocean Engineering. 2012. Vol. 42. P. 26–34. DOI: https://doi.org/10.1016/j.oceaneng.2012.01.012
3. Tu T.N. Numerical simulation of propeller open water characteristics using RANSE method // Alexandria Engineering Journal. 2019. Vol. 58. № 2. P. 531–537. DOI: https://doi.org/10.1016/j.aej.2019.05.005
4. Wang X., Walters K. Computational analysis of marine-propeller performance using transition-sensitive turbulence modeling // Journal of Fluids Engineering. 2012. Vol. 134. № 7. Art. 071107. DOI: https://doi.org/10.1115/1.4005729.
5. Nakisa M. and A.M.J. and A.A.M. Assessment of marine propeller hydrodnamic performance in open water via CFD: Proceedings of The 7th International Conference on Marine Technology (MARTEC 2010). 2010. P. 35–44.
6. Helal M.M., Ahmed T.M., Banawan A.A., Kotb M.A. Numerical prediction of sheet cavitation on marine propellers using CFD simulation with transition-sensitive turbulence model // Alexandria Engineering Journal. 2018. Vol. 57. № 4. P. 3805–3815. DOI: https://doi.org/10.1016/j.aej.2018.03.008
7. Корнелюк О. Способы задания граничных условий и геометрии гребного винта в Flow Vision // Наука и прогресс транспорта. Вестник Днепропетровского национального университета железнодорожного транспорта. 2018. № 5(77). С. 74–81.
8. Месропян А.В., Шабельник Ю.А. О влиянии параметров сеточной модели при формировании расчётной области на примере гребного винта // Вестник Уфимского государственного авиационного технического университета. 2021. № 2(92). С. 41–47.
9. Белая А.Б., Яковлев А.Ю. Forecasting of modern propeller characteristics by OpenFoam // Marine Intellectual Technologies, 2020. № 3(49). P. 40–47.

10. Belhenniche S.E. et al. Effect of pressure pores size on hydrodynamic and hydroacoustic marine propeller performances under cavitating case // Ocean Engineering. 2024. Vol. 307. Art. 118164.
11. Rizk M.A. et al. Cavitation Predictions of E779A Propeller by a RANSE-based CFD and Its Performance Behind a Generic Hull // Journal of Marine Science and Application. 2023. Vol. 22. № 2. P. 273–283.
12. Alves Pereira F.J. et al. Description of the INSEAN E779A Propeller Experimental Dataset. 2006.
13. Ali R., Tryaskin N.V. Numerical study on effect of the turbulence initial conditions on transition flow over 2D airfoil // Proceedings of the Institute for System Programming of the RAS. 2019. Vol. 31. № 6. P. 203–214.

REFERENCES

1. Sikirica A., Čarija Z., Kranjčević L., Lučin I. Grid type and turbulence model influence on propeller characteristics prediction. *Journal of Marine Science and Engineering*, 2019, vol. 7, no. 10, art. 374. DOI: https://doi.org/10.3390/jmse7100374
2. Morgut M., Nobile E. Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow. *Ocean Engineering*, 2012, vol. 42, pp. 26–34. DOI: https://doi.org/10.1016/j.oceaneng.2012.01.012
3. Tu T.N. Numerical simulation of propeller open water characteristics using RANSE method. *Alexandria Engineering Journal*, 2019, vol. 58, no. 2, pp. 531–537. DOI: https://doi.org/10.1016/j.aej.2019.05.005
4. Wang X., Walters K. Computational analysis of marine-propeller performance using transition-sensitive turbulence modelling. *Journal of Fluids Engineering*, 2012, vol. 134, no. 7, art. 071107.
5. Nakisa M. and A.M.J. and A.A.M. Assessment of marine propeller hydrodynamic performance in open water via CFD: Proceedings of The 7th International Conference on Marine Technology (MARTEC 2010). 2010. P. 35–44.
6. Helal M.M., Ahmed T.M., Banawan A.A., Kotb M.A. Numerical prediction of sheet cavitation on marine propellers using CFD simulation with transition-sensitive turbulence model. *Alexandria Engineering Journal*, 2018, vol. 57, no. 4, pp. 3805–3815. DOI: https://doi.org/10.1016/j.aej.2018.03.008.
7. Kornelyuk O. Methods of specifying boundary conditions and propeller geometry in Flow Vision. *Science and progress of transport. Bulletin of the Dnepropetrovsk National University of Railway Transport*, 2018, no. 5(77), pp. 74–81. (In Russ.).
8. Mesropyan A.V., Shabelnik Yu.A. On the influence of grid model parameters in the formation of the computational domain using the example of a propeller. *Bulletin of the Ufa State Aviation Technical University*, 2021, no. 2(92), pp. 41–47. (In Russ.).
9. Belaya A.B., Yakovlev A. Yu. Forecasting of modern propeller characteristics using OpenFoam. *Marine Intellectual Technologies*, 2020, no. 3(49), pp. 40–47. (In Russ.).
10. Belhenniche S.E. et al. Effect of pressure pores size on hydrodynamic and hydroacoustic marine propeller performances under cavitating case. *Ocean Engineering*, 2024, vol. 307, art. 118164.
11. Rizk M.A. et al. Cavitation Predictions of E779A Propeller by a RANSE-based CFD and Its Performance Behind a Generic Hull. *Journal of Marine Science and Application*, 2023, vol. 22, no. 2, pp. 273–283.
12. Alves Pereira F.J. et al. Description of the INSEAN E779A Propeller Experimental Dataset. 2006.
13. Ali R., Tryaskin N.V. Numerical study on effect of the turbulence initial conditions on transition flow over 2D airfoil. *Proceedings of the Institute for System Programming of the RAS*. 2019, vol. 31, no. 6, pp. 203–214.

ИНФОРМАЦИЯ ОБ АВТОРАХ | INFORMATION ABOUT THE AUTHORS

Али Рами Мамдух – кандидат технических наук, доцент, Санкт-Петербургский государственный морской технический университет (Санкт-Петербург, Российская Федерация),
✉ ramimamdouhali@gmail.com
Rami Mamdouh Ali, Candidate of Technical Sciences, Associate Professor, Saint Petersburg State Marine Technical University (Saint Petersburg, Russian Federation).

Ризк Мохаммед Аднан – магистрант, Санкт-Петербургский государственный морской технический университет (Санкт-Петербург, Российская Федерация),
✉ mohammed1995rizk@gmail.com, https://orcid.org/0000-0002-5084-1129
Mohammed Adnan Rizk, Master's Student, Saint Petersburg State Marine Technical University (Saint Petersburg, Russian Federation).

Статья поступила в редакцию / Received: 12.11.2024.
Доработана после рецензирования / Revised: 11.02.2025.
Принята к публикации / Accepted: 23.06.2025.